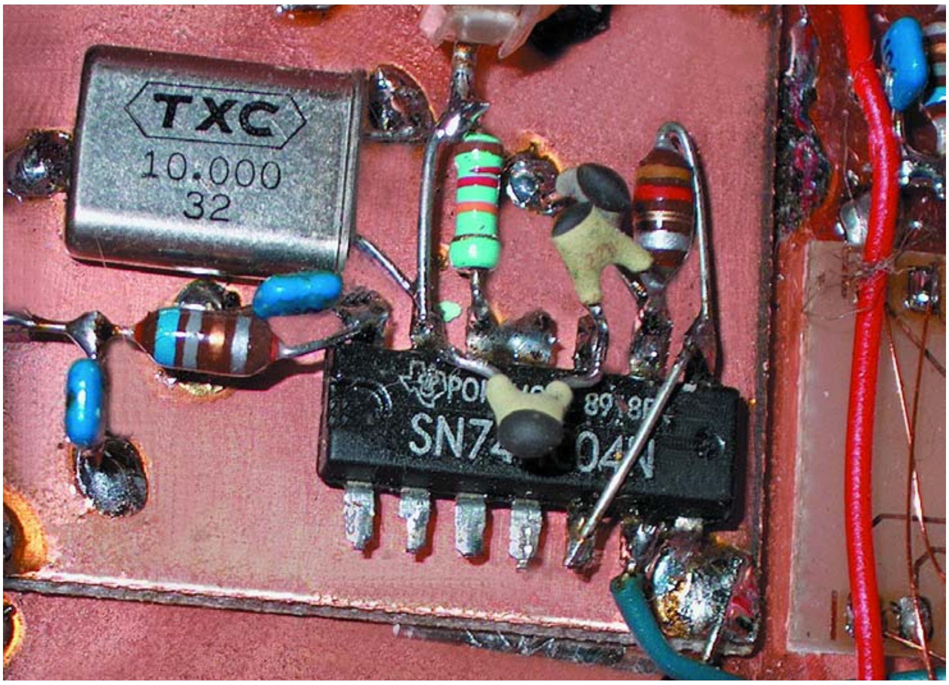
Building RF Circuits

Try the Manhattan method

By G. Baars

It’s not essential to use a printed circuit board when building RF circuitry. There is an alternative method, in which a piece of unetched circuit board material is used as a foundation for constructing the circuit. In certain situations, this method is even preferable to using a regular PCB, particularly when high frequencies are involved.



With what radio amateurs like to call the ‘Dead Bug’ construction method (which is also called ‘Ugly Construction’), the components making up the circuit are simply soldered to each other ‘suspended’ in the air, with the copper surface of a printed circuit board serving as a ground plane. Solder islands can be created at strategic locations by milling away the surrounding copper foil, or by cutting small pieces of PCB material and attaching them to the ground plane with crazy glue or double-sided adhesive tape. This avoids the need to design a printed circuit board, which means that you can work faster and still achieve amazingly good results. This type of construction is particularly handy for prototypes, since it’s easy to make changes to the circuit and components when the Dead-Bug method is used.

However, RF signals sometimes behave in their own particular manner, and this must be taken into account. A number of rules are thus indispensable in order to avoid undesired effects. This article describes how the Dead-Bug construction method can be optimised for RF design and points out a number of essential details. The result is a component mounting technique called ‘Manhattan’ for reasons which are quite obvious!

Preparation

To start with, you have to saw a suitably sized piece of PCB material. It should not be too large, but it should certainly not be too small. Next, the board must be thoroughly cleaned. This can be done using a scrubbing sponge with lukewarm water and a scouring agent, such as Cif or Vim, but a bit of steel wool or a piece of fine sand paper can also work wonders (see **Figure 1**). Some companies sell a ‘Poliblock’ for this purpose, and it also gives excellent results.

Once the board has been thoroughly cleaned, it can be silver plated if desired. You can buy liquid silver-plating preparations that only have to be lightly rubbed on using a

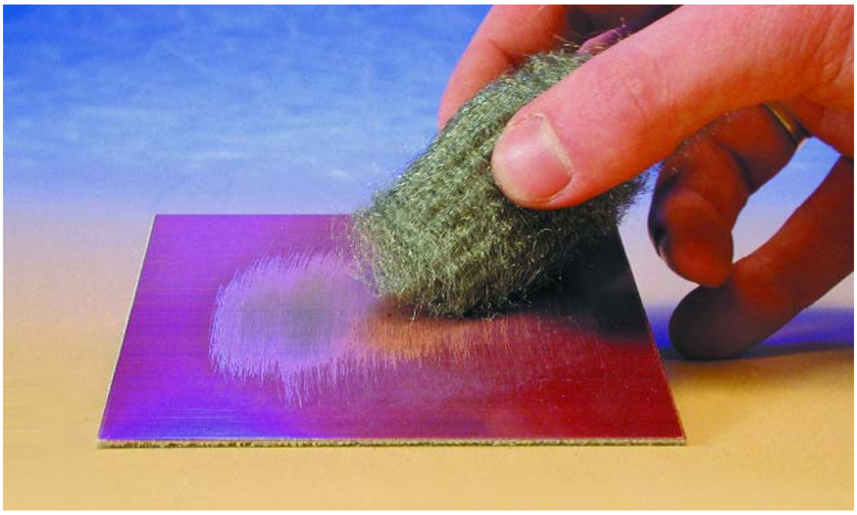


Figure 1. The printed circuit board material must always be scrubbed to obtain a clean surface.

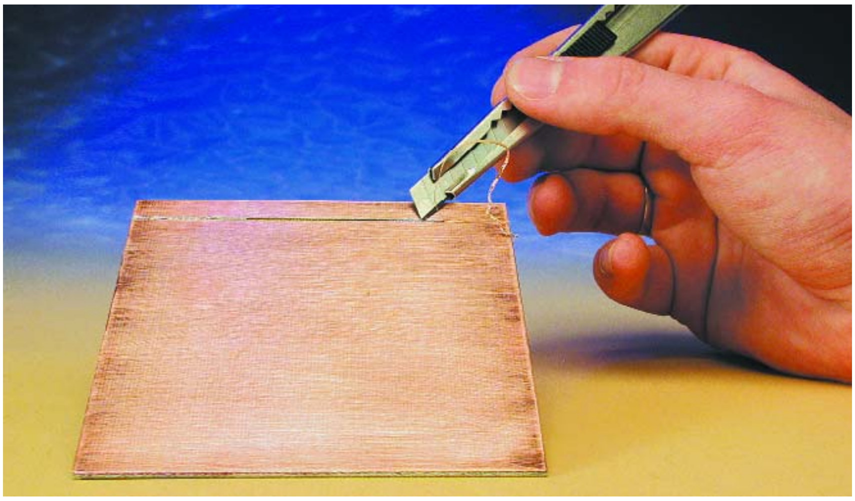


Figure 2. A small portion of the board can be isolated from the rest using a hobby knife or a router, so that it can be used as a supply rail or for some other purpose.

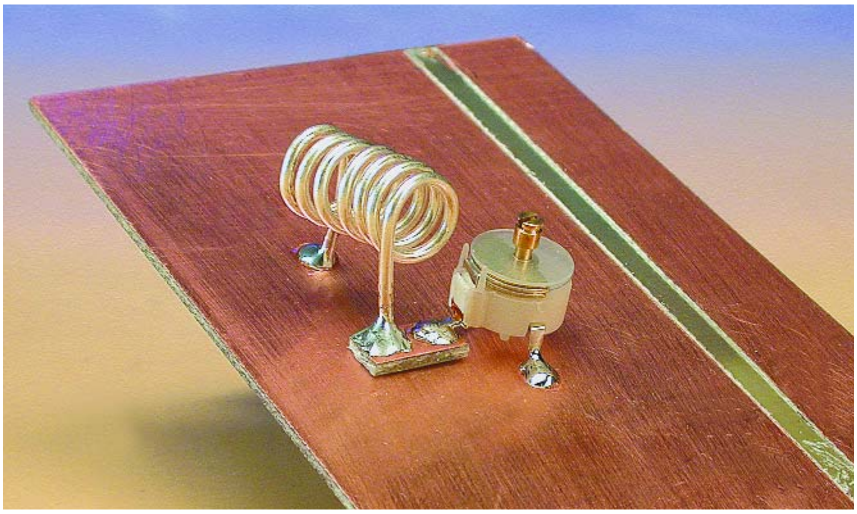


Figure 3. Coils must always be elevated a few millimetres above the board. The 'hot' end can be attached to an island.

cotton bud. This produces a thin silver coating on the surface of the circuit board, which can later be used for soldering in the usual manner. However, silver plating is by no means essential, especially if very high frequencies are not involved.

On the other hand, it is a good idea to coat the circuit board with a layer of PCB lacquer. This keeps the board nice and shiny, so that it can still be soldered even years later.

It's sometimes handy to isolate an edge of the board from the rest. The isolated portion can then be used as a supply rail. An edge can be readily isolated by making two parallel cuts in the copper foil with a knife and then prising out the copper between the cuts (see **Figure 2**). An easier method is to use a minidrill with a small routing bit.

The real work

Now you can start building. If you have a schematic diagram, it's best to start with the input portion, which is usually located at the left-hand side of the drawing. If the first stage is an RF input amplifier, for instance, with its input and output tuned using trimmers and air-core coils, it is advisable to fit the coils at right angles to each other. This minimises the coupling between the coils, thus reducing the tendency of the stage to oscillate.

In such cases, a small screen is often fitted above the amplifier between the input and output circuitry, in order to eliminate feedback. It can be made from a piece of PCB material around 2 to 3 cm high, which is soldered directly to the ground plane.

Self-supporting coils are best made from silver-plated copper wire with a diameter of approximately 1 mm. If the 'cold' end of the coil is connected to ground, it can be soldered directly to the ground plane, which provides adequate mechanical strength. If one end of the coil must be connected to the supply rail or something other than ground, an island can be created using a small piece of PCB material or some other method, in order to adequately support the coil. A trimmer or other component can then be soldered to this island as well (see **Figure 3**). Naturally, if you have a minidrill and a small router bit, you can simply rout out an island where you need it.

While we're on the subject of trimmers, whenever you use a foil (PTFE) trimmer, you should always rebend the middle lead at a right angle so it can be used a solder point for another component, such as the 'hot' end of a coil. The other two leads can then be soldered to the ground plane (for example). If these two leads are not connected to ground, a small island can again be used to support the trimmer.

Short connections

When working with RF circuits, it's always a good idea to keep signal-carrying connections as short as possible. Good decoupling is also necessary, and it is important to solder the decoupling capacitors as close as possible to the transistors or ICs carrying the signal. Small ceramic capacitors are preferable, since they allow the connections to ground to be kept short.

You should bear in mind that a length of wire has an intrinsic self-inductance of around 10 nH per centimetre. At relatively high frequencies, this inductance can easily become significant if adequate decoupling is not present. Decoupling also prevents feedback via the supply rail, which can cause oscillations and other side effects. At very high frequencies, it is advisable to use SMD capacitors for decoupling. They have better characteristics than other types of capacitors, such as multilayer (film) capacitors.

With the dead-bug method, it is also highly advisable to keep the circuitry fairly low and close to the surface of the PCB material, which serves as a ground plane. This shorts out possible interference fields, thus reducing the likelihood of radiation and feedback. However, coils must always be fitted a few millimetres above the surface of the board, since otherwise their fields will be short-circuited (which reduces their self-inductance and increases damping). This applies to DIY coils as well as ready-made E12 miniature chokes.

Rigid construction

Another factor that plays a role in certain circuits is mechanical strength. For instance, a VFO (variable frequency oscillator) circuit must be built in a mechanically rigid manner in order to avoid detrimental effects on its frequency stability. There must be no noticeable effect from a shock or blow. For this reason, special attention is often given to the resonant circuit of a VFO. For instance, the coil is often wound on a ceramic form, and if the coil is small, heavily silver-plated copper wire is preferred. VFOs are also commonly fully screened using sheet-metal enclosures, with the advantage that the amount of electromagnetic radiation is reduced and the circuit is screened from air movements. This is important, since moving air may cause undesired temperature changes that can produce frequency fluctuations.

Fitting semiconductor devices

Various methods can be used to fit ICs. Some builders prefer to mount them flat on their backs with their leads in the air, and then solder all the other components to the leads. The disadvantage of this method is that the arrangement of the IC leads is a mirror image of the usual arrangement, which can be somewhat inconvenient in practice.

Another method is to rebend all of the leads of the IC (except the leads connected to ground) at right angles to the package and then clip off the narrow ends of the leads. This produces convenient solder islands, as can be seen in the title illustration for this article. The easiest way to bend the leads is grasp them ten at a time using flat-jawed pliers. All of the wide portions of the leads must end up being perpendicular to the package once they have been rebent.

Working with SMD devices is somewhat more difficult, but certainly not impossible. It is often possible to directly solder to the leads of an IC with eight pins at a 50-mil pitch, for example, but if more pins are involved or a finer pitch is used, it is recommended to make an adapter board. Such an adapter board can convert the SMD lead pitch to a standard DIP pitch with 100-mil spacing, which is easy to solder to.

As a rule, transistors can be best placed flat against the circuit board, as illustrated in **Figure 4**. The outer leads are bent at right angles to the left and the right, so that they act as 'solder pillars' for the following components. Voltage regulators in the common TO220 package can also be quite readily fitted flat against the surface of the board, which provides the extra benefit of improved cooling.

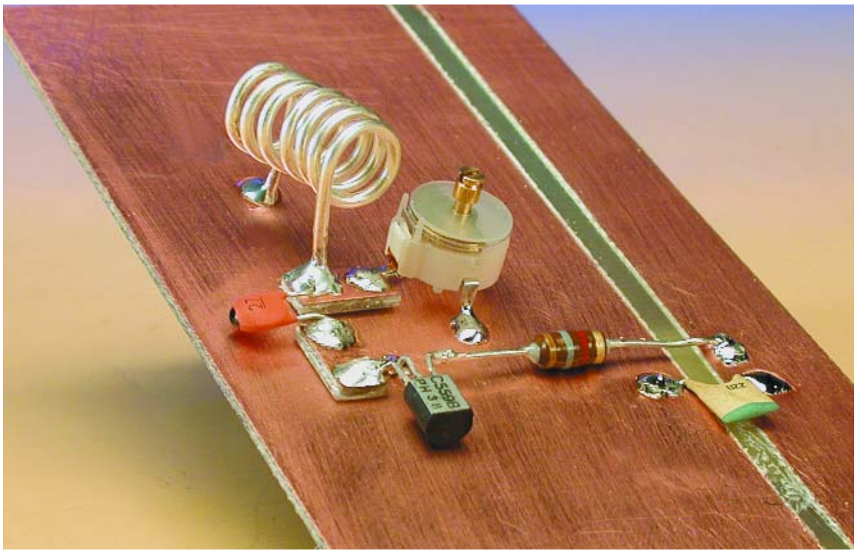


Figure 4. With a few homemade islands, a complete circuit can be built quite quickly using the Manhattan method, which may be described as an RF-optimised variant of Ugly Construction.

In the case of RF power amplifiers, the transistors that are used often have rather wide leads. They can be supported using (you guessed it!) small islands sawn from PCB material and attached to the circuit board, which can also serve as solder points for other components. It is also possible to cut or rout islands in the copper foil of the base circuit board, as previously described. This is a difficult job with a hobby knife, but with a minidrill and a small routing bit it's a piece of cake.

Conclusion

If there are connections that carry heavy currents, such as a supply connection or a loudspeaker connection, it is preferable to use a 'star' grounding point. The advantage of using star points is that the connections in question are individually routed to the central grounding point. This prevents the generation of undesired ground currents that could have detrimental effects on the circuit.

(030032-1)